SELECTED ENGINEERING PROBLEMS
NUMBER 4
INSTITUTE OF ENGINEERING PROCESSES AUTOMATION
AND INTEGRATED MANUFACTURING SYSTEMS

Piotr GENDARZ, Dominik RABSZTYN*

Institute of Engineering Processes Automation and Integrated Manufacturing Systems,
Faculty of Mechanical Engineering, Silesian University of Technology, Gliwice, Poland
*dominik.rabsztyn@polsl.pl

OPTIMIZATION OF THE DIMENSIONS VALUES BY MEANS OF FINITE ELEMENT METHOD

Abstract: The paper presents the possibilities of applying the CAE software based on the Finite Element Method to optimize the dimensions of the machine elements.

1. Introduction

Due to the dynamic technological progress the product that was just recently introduced to the market, soon becomes out of date. Nowadays the market faces the producers with a difficult task: they have to develop a product that meets the specific requirements of customers, has a competitive price, high quality and durability while on the other hand they have to reduce the duration of the design and production process. The application of computer-aided optimization process combined with simulation helps to meet the requirements. Optimization of the dimensions by means of the Variational Analysis based on the Finite Element Method is one of the most important stages of the design process which aims at improving the design characteristics of the future product as well as reducing the time of introducing the new product to the market.

2. Description of the method of optimizing the dimensions values by means of the variant analysis

In the case of a conventional material strength analysis with the usage of the Finite Element Method, all of the design features (geometric parameters, material strength properties, size of finite element, etc.) are fixed values. Therefore, the matrix of the model stiffness is constant for a given set of parameters. The obtained results thus illustrate how the analyzed element will behave a specific set of parameters, without any information on the possibility of introducing variant changes to improve the properties of the future product. One analysis allows obtaining only one point in the space of possible solutions, which is usually far from the optimal solution. Manual changes in parameters of the model and further analyses is time consuming and does not guarantee getting close to the optimal solution.
By applying the Variational Analysis implemented in programs supporting the design process of such as I-DEAS (Variational Analysis) [2] or NX (Geometry Optimization) [1,3] it is

possible to obtain optimal values of the dimensions of the elements in a relatively short period of time according to specific criteria such as:

- not exceeding the allowed stresses
- not exceeding the defines transformations,
- minimal mass or volume of the element.

In the Variational Analysis the design space (parameterized CAD model) is transferred directly to the analysis, so that the software automatically changes the values of the defined dimensions and conducts an analysis for each of the variants. The stiffness matrix is therefore a function of changed optimization parameters. The variant space of the model can contain unknowns such as geometrical data (the values of variable dimensions) or material properties. This results in not only an optimal solution (with optimally chosen geometrical and material parameters), but also shows how the strength of the component will change for the whole range of design variables.

3. Stages of the variant analysis

The process of optimizing the dimensions with the Variational Analysis has been divided into three main stages: the Preparatory Stage, Sensitivity Analysis and Parametric Analysis.

The Preparatory Stage (Fig. 1) includes the following oparations [2]:

- developing a parameterized CAD model
- defining the material of the element,
- generating a finite element mesh model,
- defining the boundary conditions,
- determining the loads,
- initial strength analysis.

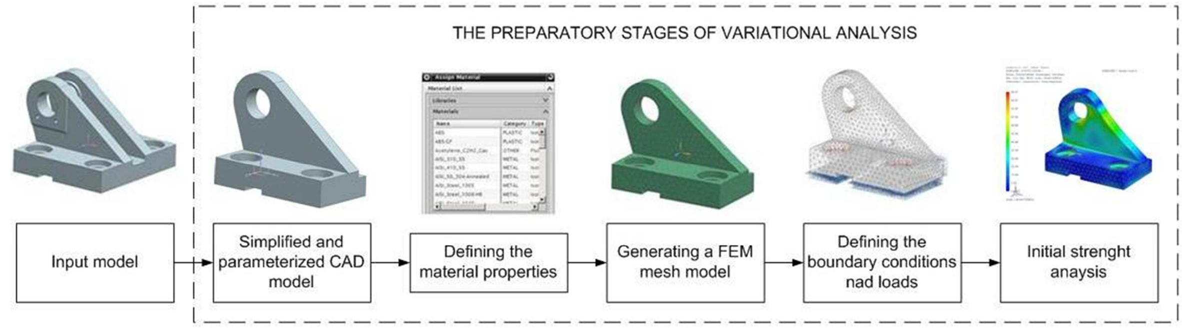
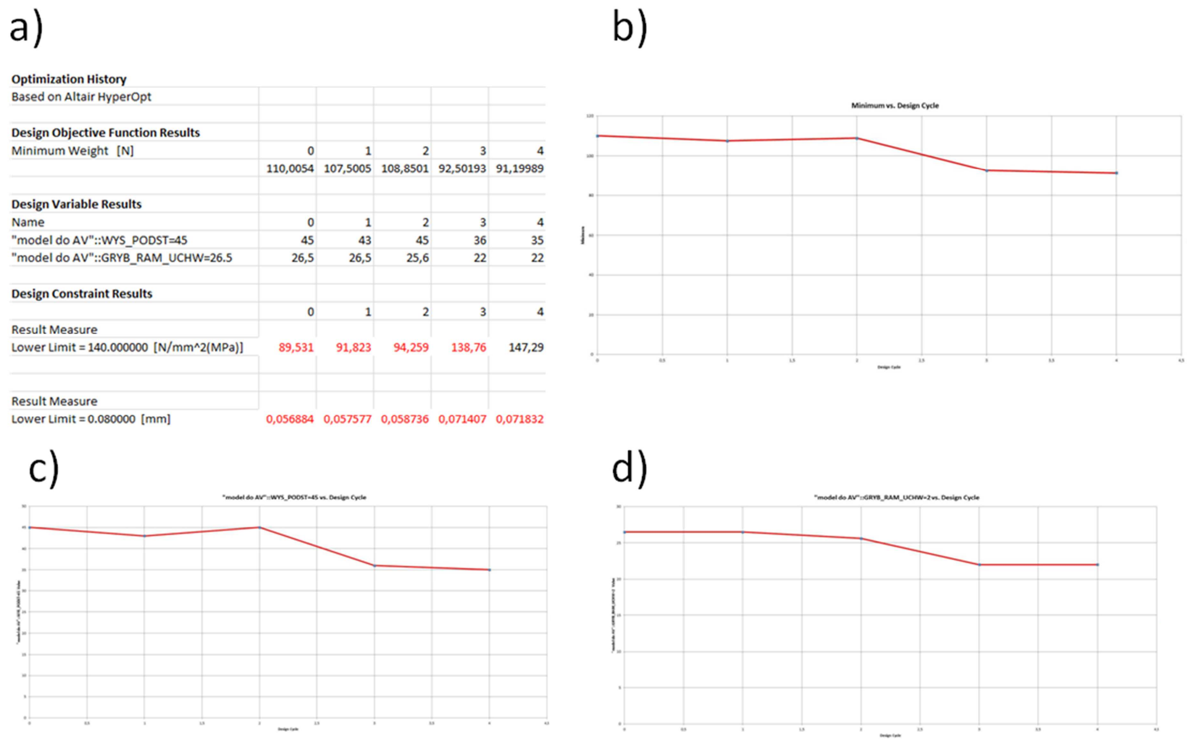


Fig.1. Preparatory Stages of the Variational Analysis

The Sensitivity Analysis determines the relations between the variable parameters (geometrical and material) and the defined criteria. It allows identifying the parameters most influencing to change the results of the stress analysis. The result of the analysis is the histogram of the sensitivity of the dimensions on certain criteria.

In the parametric analysis diagrams presenting the changes in dimensions values are generated (Fig. 2) depending on the conducted iteration. Before starting the analysis, it's necessary to define objective function and criteria of optimization and geometrical parameters subjected to the optimization process. These parameters are selected based on the results of

Sensitivity Analysis. It is also necessary to define the limit number of iterations and the ranges of variation of selected parameters.



Optimization History					
Based on Altair HyperOpt					
Design Objective Function Results					
Minimum Weight [N]	0	1	2	3	4
	110,0054	107,5005	108,8501	92,50193	91,19989
Design Variable Results					
Name	0	1	2	3	4
"model do AV"::WYS_PODST=45	45	43	45	36	35
"model do AV"::GRYB_RAM_UCHW=26.5	26,5	26,5	25,6	22	22
Design Constraint Results					
	0	1	2	3	4
Result Measure					
Lower Limit = 140.000000 [N/mm^2(MPa)]	89,531	91,823	94,259	138,76	147,29
Result Measure					
Lower Limit = 0.080000 [mm]	0,056884	0,057577	0,058736	0,071407	0,071832

Fig. 2. Results of the optimization analysis for individual iterations: a) values of the variable parameters and the optimization criteria, b) diagram of the element's mass, c) d) values of the optimized dimensions

Based on the analysis of variability of stresses, strains, and mass, the algorithm determines the optimal values for the dimensions, which are automatically transferred to the CAD model. The relational - graphic parameterization automatically updates the initial model to the form with the optimal value dimensions. The Fig. 3 presents the comparison of the strength analysis of a fragment of a bracket before and after the optimization of its dimensions.

For the presented in the paper example it was possible to reduce the weight from 22.2kg to 18.6 kg by increasing the strain of material from 90MPa to 139MPa, and not exceeding the allowed displacements.

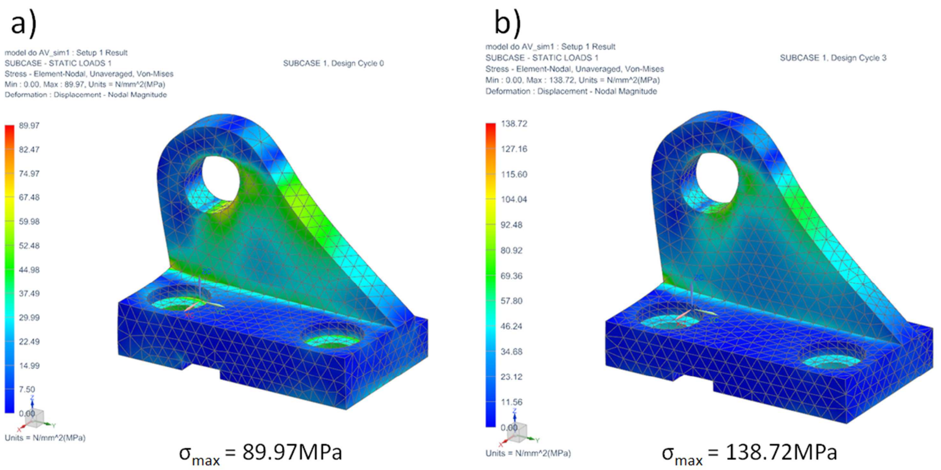


Fig. 3. Distribution of stresses: a) model before optimization, b) model after optimization with the Variational Analysis

The optimal design due to the defined criteria should be further subjected to the process of standardization of dimension values and be verified regarding the criteria of the identity of the coupled dimensions.

4. Conclusion

In the example of geometrical dimensions optimization of a bracket presented in the paper the minimum weight and the following boundary conditions were determined as the main criteria of the optimization:
- stresses in the element less than 140MPa ($\sigma_{max} \leq 140$MPa)
- displacement of any point of the yoke less than 0.1 mm.

The optimization was performed using the Geometry Optimization tool, implemented in NX8.5 CAD/CAE software.
As a result of optimization using Variational Analysis the weight of the bracket has been reduced by 16% with the defined boundary conditions not being exceeded.

The application of the optimization of dimensions (geometrical and material) by means of Finite Element Method not only improves and accelerates the design process, but also leads to a reduction the weight of element, allows better use of the material and shortens the duration of the manufacturing process. These savings will be directly reflected in a lower final price of the product and/or the manufacturer's financial gain.

References

1. Gendarz P.: Flexible modular systems. Gliwice: Publisher of Silesian University of Technology, 2009 (in Polish).
2. Cielniak M.: Optimization using FEM in NX, STEEL Metals & New Technologies, July August 2012, pp. 63-66 (in Polish).
3. Gendarz P., Rabsztyn D.: The exemplar design features in creation of the series of types, Journal of Achievements in Materials and Manufacturing Engineering 2012, Vol. 55/2, pp. 488-497.